Date:

* Car Crash Analysis Project

It cost around 2500$, a year of research to perform an actual crash Analysis

Problem → Perform a car crash Analysis for a car Running at 60 km/hr

Steps:-

Importing the CAD

Altair Hypermesh
↓
View Profile → LS-Dyna → OK
↓
Import CAD → select SolverDeck
→ select (basemodel.k) File
→ OK
→ Import

#Setup for Analysis

↓
View Mesh → Shaded element on bottom bar
↓
View → Click solver browser #Use to setup the Analysis
↓
Entities → Right Click → Create → Section → Section cut

Name	Value
NIP = No of integration point	= 4
T1 = value of thickness	= 5mm

#Assigning the Material

→ Right click → Create → MAT → MAT(1-50)
→ MAT(Power law Plasticity)
Name as Aluminum

density,	Rho	1.4e-09
	E	75000
Poisons ratio,	PR	0.33
	K	0.097
	N	0.33587

Yield Stress, SIGY Hoompa

Date:

Ineat → MAT → MAT(I-SO) → MAT(RGD)

"name rigid"

Rho - 7.85e-09

E - 2.1e5

PR - 0.3

#Now we will assign section & material to the parts.

Part → Part → Chassis → Rho

Property → Shell 1

Material → Aluminium

→ Wall

Property → Shell 1

Material → Rigid

#Now Specifying Boundary conditions for Ineat Analysis

Wall will be rigid.

Initial velocity of car will be 60 Km/hr

We will also specify the contact between the wall & chassis, to capture the interval between this par

Date:

Entities → Right click → Create → BOUNDARY

BOUNDARY.SPC_NODE

#Creating nodes of wall

NODE → by path ("to select all node on edge of wall to create single point condition")

Select corner node of wall in CAD

→ ☑ check all spc degree of freedom

→ Create

→ return

#Creating node for chassis

Create → Set → Set_NODE → Set_Node_list →

"Name - Chassis Node"

Entity IDs → Node → node → by collector

☑ chassis → Select

#Giving the velocity to the car

Create → Initial → Initial velocity. Generate

Name - "velocity"

Options → STYP → Node set ID

NSID → Chassis_nodes

VZ → -16666.0

To define interaction between wall & chassis

Create → Contact → Contact_Automatic surface to surface

Name "Contact"

Slave entity SSID → Components → Chassis

Master entity MSID → Component → Wall

FS = 0.2, Static friction ID

FD = 0.1, Dynamic friction ID

Date:

#To Specify total run time for Analysis

Create → Control → Control_termination

ENDTM → 0.15, end time

Create → Control → Energy

HGEN → Option 2

RWEN → Option 2

SLNTEN → Option 2

RYLEN → Option 2

} Energy dissipated by all forms

#To output the results for post processing

Create → Database → Binary_D3 Plot

DT = 0.0015, no of time intervals

#To output Global variables (Work & Energy)

Create → Database → Database_Option

GLSTAT ☑, global statistics

DT → 0.0015

MATSUM → DT → 0.0015

(Printout numerical values)

"Analysis setup is completed

File → Export → Solver Deck → Create new file → save as "run1" → Export

"The K file is now ready for run"

Date:

#We will use this 'K' file to load & solve it in LS Dyna solver

#After the Analysis we will view it in hyperview

Open LS Dyna Program Manager

Start ↓ → Select the 'K' file → Set CPU option as per need → Click on RUN.

Open hyperview to view the Results of crash Analysis

Select d3plot file from working directory

* Result.

Displacement Results → → Apply → 4.000E+02

Stress Result → → Stress → Averaging Method → Simple → Apply

Energy Conservation. Plot → Split graphic window → → Hypergraph2D

load glstat file

Select Energy → KE

IE

TE